Proceedings of FEDSM2005
2005 ASME Fluids Engineering Division Summer Meeting and Exhibition
June 19-23, 2005, Houston, TX, USA

FEDSM 2005-77225

NAVIER-STOKES SIMULATION OF AIR-CONDITIONING FACILITY OF A LARGE MODERN COMPUTER ROOM

Jasim U. Ahmad
ELORET Inc.
NASA Ames Research Center, M/S T27B, Moffett Field, CA 94035
Tel: (650) 604-4473, E-mail: ahmad@nas.nasa.gov

Shishir A. Pandya, William M. Chan, Neal M. Chaderjian
NASA Ames Research Center, Moffett Field, CA 94035

Abstract

A 3-D full Navier-Stokes simulation of a large scale computing facility at NASA Ames Research center was carried out to assess the adequacy of the existing air handling and conditioning system. The flow simulation of this modern facility was modeled with a viscous, compressible flow solver code OVERFLOW-2 with low Mach number pre-conditioning. A script was created to automate geometry modeling, grid generation, and flow solver input preparation. A new set of air-conditioning boundary conditions was developed and added to the flow solver. Detailed flow visualization was performed to show temperature distribution, air-flow streamlines and velocities in the computer room.

Introduction

NASA recently assembled one of the world's fastest operational supercomputers to meet the agency's new high performance computing needs. This large-scale system, named Columbia, consists of 20 interconnected SGI Altix 512-processor systems, for a total of 10,240 Intel Itanium-2 processors. High-fidelity computational fluid dynamics (CFD) simulations were performed for the air flow in the NASA Advanced Supercomputing (NAS) computer room at NASA Ames Research Center. The purpose of the simulations was to assess the adequacy of the existing air handling and conditioning system and make recommendations for changes in the design of the system if needed.

Appropriate cooling system design for such a large facility is very important as the equipment dissipates a significant amount of heat. There are only a handful of publications available that utilize CFD methodology to enhance the design phase of the air-conditioning system. Most important among them is a recent study by Schmidt et. al (2001) which uses a 2D model with depth averaging continuity and momentum equations. Their study combined experimental work and numerical modeling. Other CFD studies with limited scope and applications are discussed in the above mentioned reference.

The present simulation solves the full 3-D Navier-Stokes equations for the room with all major components including their detailed geometric complexity. The simulations were performed with NASA's OVERFLOW-2 CFD code (Buning et. al 2003, 2004) which utilizes overset structured grids to model the various pieces of hardware present in the room. The geometry modeling and grid generation were handled by the NASA Ames software package Chimera Grid Tools (CGT) (Chan 2002, 2004). A new set of boundary conditions was developed and added to the flow solver for modeling the room's air-conditioning and equipment cooling systems. Boundary condition parameters for the flow solver are based on the cooler design flow rate (CFM) and some reasonable assumptions of flow and heat transfer data for the floor and central processing units (CPU).

The NAS facility's main computer room contains a subfloor below the main floor. Several International Standard Unit (ISU) coolers located along the room walls supply cool air to the subfloor from which the cool air is drawn into the main computer room through perforated floor tiles strategically placed along CPU racks. Different cooling tile perforations influence the amount of airflow into the main floor. The mass flow distribution of these cooling floor tiles requires a coupled simulation of the flow for both the main computer room and the subfloor. Due to the urgency of finishing the simulations in a short time, a full 3-D simulation was not performed for the subfloor. However, a quick 2-D simulation with OVERFLOW-2 of an isolated cross-section of the subfloor was performed to assess the mass flow distribution between the ISU coolers and perforated tiles. From this study it was concluded that it is reasonable to assume that the flow rate is evenly distributed among all the perforated floor tiles. The work by Schmidt et. al (2001) came to a similar conclusion.

The compute nodes (see Fig.1) are grouped in pairs of racks with an aisle in the middle. High-speed connection cables connect the racks with overhead cable trays. The cool air from the cooling units is pumped into the sub-floor. The CPU cooling fans draw cool air from the subfloor through the perforated floor tiles, which run along the outside length of each rack, and eject warm air into the center isle between the pair of racks. This outside length along the rack will be referred to as the 'cool side' and the center-isle side along the length of the rack will be called the 'warm side' for the remainder of this paper. The warm air is eventually drawn into the coolers located along the walls of the room. These coolers are not evenly distributed along the room walls.

One major concern is that the hot air ejected into the middle isle might recirculate back into the cool side of the rack and cause thermal short-cycling, which can result in inadequate CPU cooling. Another related problem similar to CPU-rack short cycling is a cooler induced short cycling. There are two air intake openings for the ISU-coolers, one on the top of the ISU, and a second on the front of the ISU. Some of these ISU coolers are very close and run parallel to some CPU racks in the NAS computer room. It is possible that the front intake of an ISU may ingest some of the cool air from the perforated floor tiles, thus restricting the cool-air supply to the CPU racks. One remedy for this type of short cycling is to close up the front ISU air intake and increase the area of the top ISU intake by making a diverter-like extension. This proposed cooler intake modification was also modeled to assess if it provided better CPU airflow characteristics.

The simulations analyzed and addressed the following important elements of the computer room:

1) High-temperature build-up in certain regions of the room;
2) Areas of low air circulation in the room;
3) Potential short-cycling of the computer rack cooling system.

Detailed flow visualization is performed to show temperature distribution, air-flow streamlines and velocities in the computer room.

Geometric Modeling, Grid Generation, and Flowsolver Preprocessing

The computer room at the NAS facility is quite large as it accommodates 10,240 processors. The current simulation models an L-shaped partition of the room which contains about 60% of the processors and all the mass storage units. Besides the CPU racks, there are various other components like Power Device Units (PDU), cable trays, and a part of the NAS computer control room. These various components are shown in Fig. 1. The ISU coolers are located along all sides of the room walls.

A script was developed based on the script library procedures from the Chimera Grid Tools software package to automate the geometry modeling, grid generation and flow solver input preparation processes. This script automatically creates the geometry, and structured overset viscous surface and volume grids for all the main components in the computer room. Features model include the CPU racks, disk racks, overhead cable trays, power units, cooler units, control room, and mass storage system. The dimensions and locations of these components, as well as the grid spacings and stretching ratios of the grids, are parameterized in the script. With any parameter change, a new grid system can be regenerated in just a few minutes on a desktop computer.

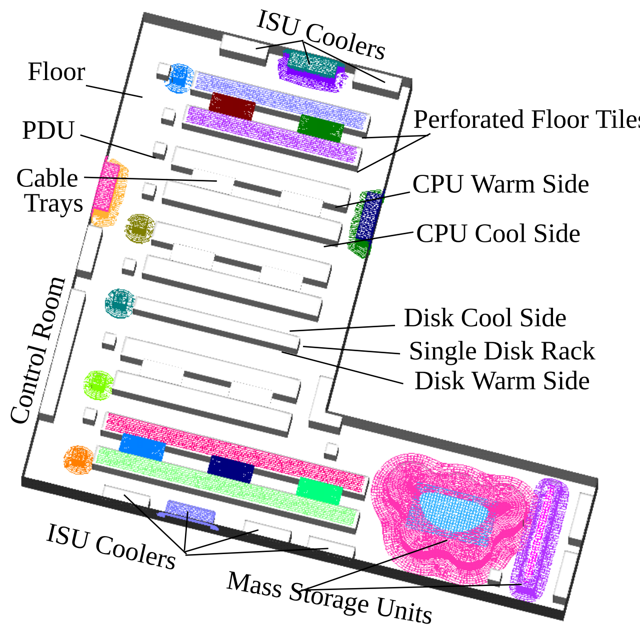

Figure 1. Room Geometry and Boundary Surfaces.

For flow solver pre-processing, the script generates the hole cutting instructions for inter-grid connectivity (Meakin 2001), and the input file for the OVERFLOW-2 flow solver which includes all the boundary conditions. The boundary condition types (viscous wall, air intake or exit) and where they are applied are also parameterized in the script. With the high level macros available in the CGT script library, the entire script was developed in about 1 to 2 man weeks, with a basic version completed in just 3 man days.

Governing Equations

The governing equations in the OVERFLOW-2 code are the full 3-D Navier-Stokes equations in conservative form for the solution of compressible viscous flows. They are written as

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = L(Q) \quad (1)$$

where $Q = [\rho, \rho u, \rho v, \rho w, e]^T$ is the state vector, E, F, G are the invisicid fluxes and $L(Q)$ is the viscous operator. Since the flow inside such a facility is of very low speed, low Mach number pre-conditioning is utilized in the OVERFLOW-2 computation for improved stability and accuracy of the numerical model. Details of this pre-conditioner and other related topics about the algorithm can be found in Pandya et. al

(2003) and Buning et. al (2003). The Spalart-Allmaras (1992) turbulence model is used in these computations.

Boundary Conditions

In order to properly prescribe the boundary conditions for such a facility with a subfloor underneath the main floor, one needs to simulate the subfloor and the main floor together. The subfloor's air distribution decides how much air will be supplied through the different floor tiles. The subfloor has many components such as pipe lines and cable trays that can produce blockage. Also, the cooling units are located in a non-uniform manner along all the surrounding walls of the room, and they deliver air to the subfloor where air streams can merge coming from the different ISUs. Moreover, the tile openings to the main room have different perforations along its length. However, for the present study, in order to avoid additional complexity and longer setup time, boundary condition parameters for the flow solver are based on uniform floor tile and cooler CFM (flow rate) ratings and some reasonable assumptions of flow and heat transfer data for the floor and CPUs. These boundary conditions were developed based on our initial experience with the problem and is focused on an internal main room flow with many obstacles as well as open spaces. The boundary conditions for the various components are given below:

<u>Inflow Boundaries:</u>

<u>Floor tile:</u>
Normal component of velocity is specified based on CFM.
All the other velocity components are set to 0.
Density is extrapolated.
Temperature is specified based on cooler settings.

<u>CPU Racks 'Warm side'</u>:
Normal component of velocity is specified from the fan flow rate.
All the other velocity components are set to 0.
Pressure P is extrapolated.
Constant temperature T is specified based on the CPU fan heat transfer rating.

<u>Outflow Boundaries:</u>

<u>Cooling (ISU) unit face:</u>
Normal component of velocity is specified based on CFM rating.
All the other velocity components are extrapolated.
Pressure P and density ρ, are extrapolated

<u>CPU Racks 'Cool side'</u>:
Normal component of velocity is specified from the fan flow rate.
All the other velocity components are extrapolated.
Density ρ, and Pressure P are extrapolated

In addition to the CPU racks, there is also a single data rack in between two of the CPU rack pairs as shown in Figure 1. This disk rack's boundary condition is similar to the CPU rack with a different fan heat transfer rating.

The surfaces where boundary conditions were applied are shown in Figure 1. All other room components such as room, floor, walls, ceiling, power units, cable trays above the racks, and mass storage units have the usual no-slip adiabatic boundary conditions.

Results

These steady-state Navier-Stokes simulations were carried out with a grid size of approximately 12 million grid points. The qualitative results are presented in the form of extensive flow visualization showing room temperature distribution, flow stream line paths and air velocities, and the potential for short cycling of the CPU racks. There were no detailed experimental data to validate the computations. In fact, the grid generation and flow simulation processes were completed in 6 weeks, to help the CPU installation deadline. However, spot checks of the temperature regions of high and low speed flow in the NAS facility after the CPU installation showed trends that closely agreed with the simulation. The surface temperature distribution is shown in Figure 2.

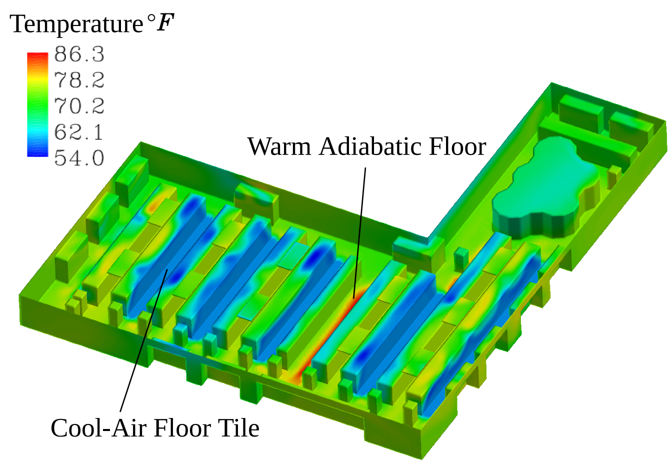

Figure 2. Room surface temperature.

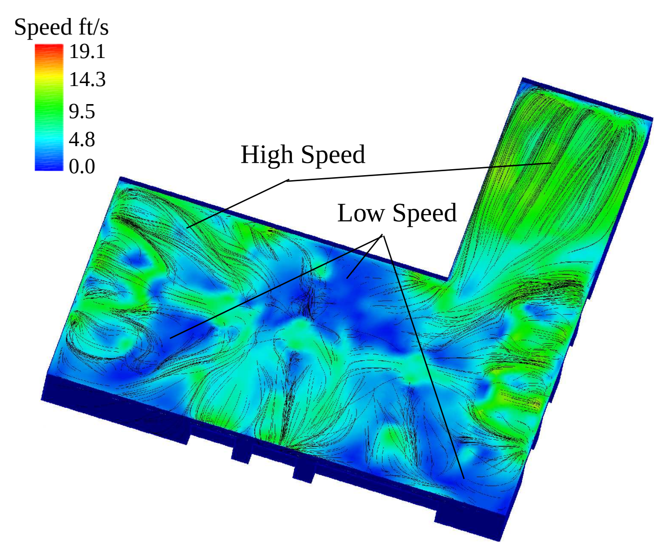

Figure 3. Air speed and flow streamlines at a height of 8.5 ft. from the floor.

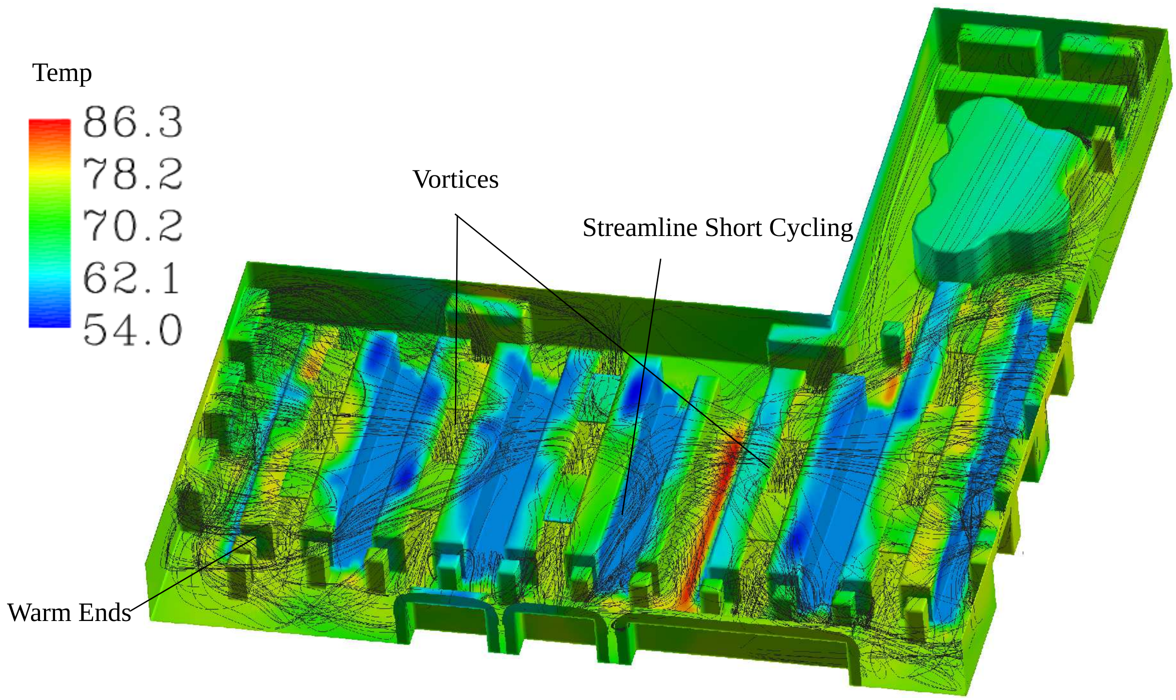

Figure 4. Temperature/flow streamlines showing short cycling and vortices.

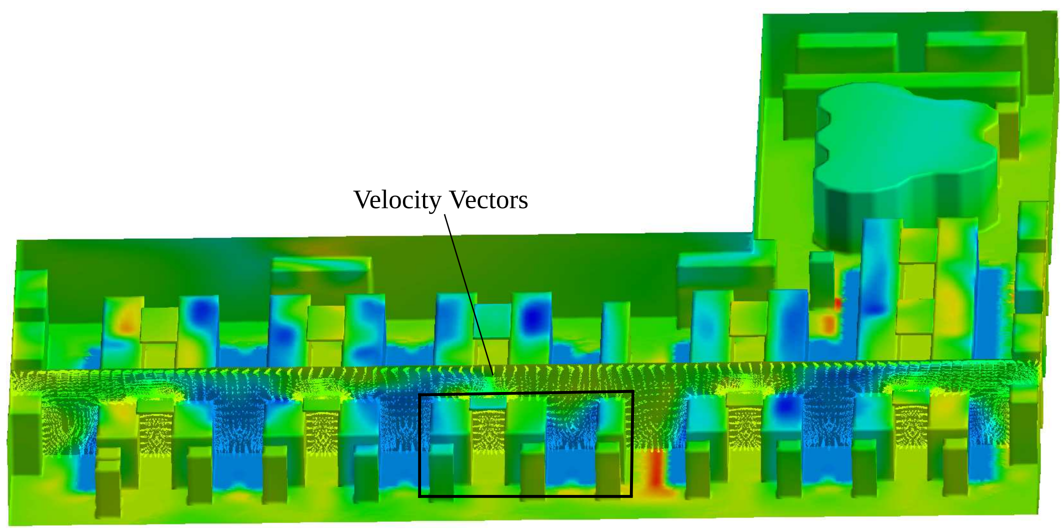

Figure 5. Temperature and velocity vectors at a section of 18 ft. from the control room wall.

The warmest spot observed in the room is on the warm side of the disk rack and a very small distance up from the floor surface. This is probably due to fact that the floor boundary was specified as a adiabatic floor rather than an isothermal surface. This will be investigated in a future study. Figure 3 shows the room air-speed distribution along with the streamlines at a height of 8.5 ft. above the room floor and close to the ceiling which is at a height of 12ft. The particles were released from various locations, specifically from the middle isle of the different racks. This identifies areas of low and high speed flows. Low speed regions are located at several spots in the room where ISU coolers are sparingly distributed, and high speed recirculated air is found near the ISU coolers. These locations of high and low speed regions vary due to the non-uniform distribution of the ISU coolers (see Fig. 1). Figure 4 shows similar flow streamlines with the temperature distribution. Particles were released from the hot rack surfaces. Small vortices are observed above the rack and near the cable trays. A closer inspection of the particles show a slight short cycling of the streamlines from the warm side to the cool side. In addition to the high temperature near the disk racks, the end around part of the CPU racks are slightly warmer as indicated.

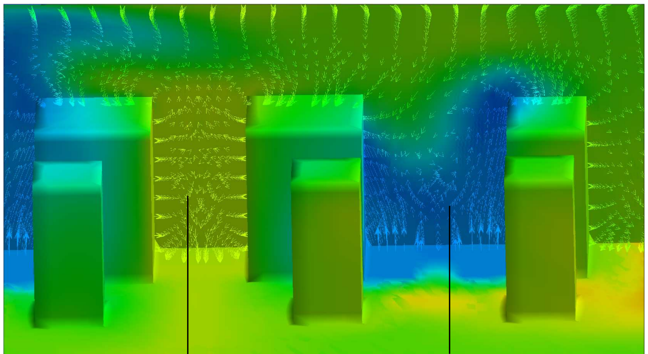

Figure 6. Temperature and velocity vectors at a section of 18 ft. from the control room wall.

Figure 5 shows the temperature distribution at a section of 18 ft. from the control room (See Fig.1). The close-up view around the middle of the room shown by the black rectangular outline in Fig. 5 is shown in Fig. 6. Almost uniform temperature distribution is shown in the middle of isle. Figure 7 shows a similar section at a distance of approximately 25ft. from the control room. The detail view shows the flow blockage due to the cable trays, and the down draft on the ‘cool side’ showing a slight short cycling.

.

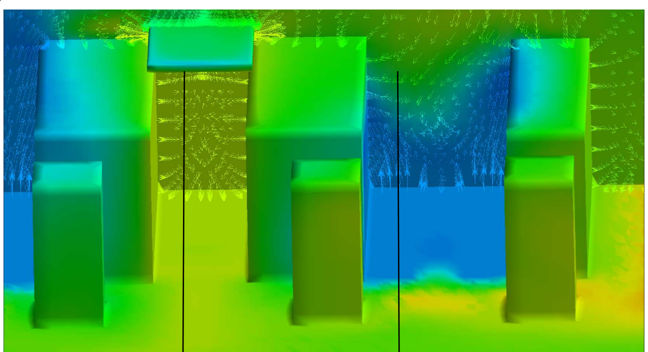

Figure 7. Temperature and velocity vectors at a section of 25ft from the control room wall.

Figure 8 represents the temperature distribution along a length of the room about 32 ft. from the control room wall and about 3 ft. above the floor. The discontinuity in the temperature plot is due to the presence of the CPU racks. It shows the ‘warm side’ and ‘cool side’ of the different racks. The geometric orientation of the different racks are depicted at the bottom of the plot (see also Fig. 1). The hot recirculated air returning to the ISU coolers are shown near the ends of the figure. The temperature distribution at the same section but just above the cable trays and at a height of 7ft. is shown in Fig. 9. At this height, there is a rapid change in temperature between the cool and warm side. Most of this temperature field is elevated, varying between 65-75 degrees Fahrenheit.

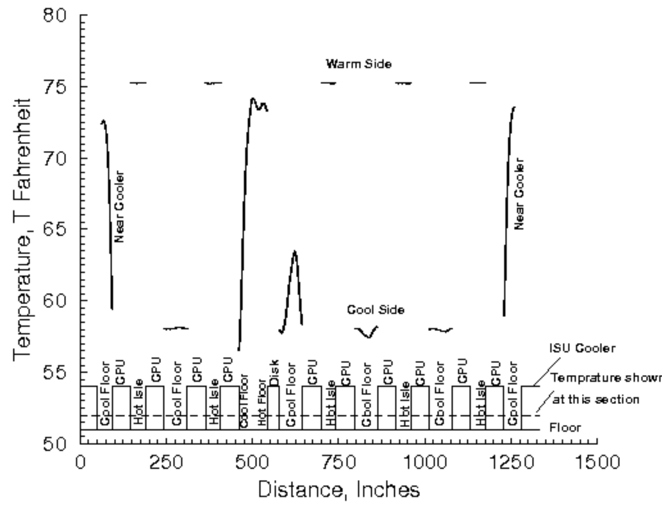

Figure 8. Temperature at a section of 32ft. from the control room wall and at a height of 3 ft.

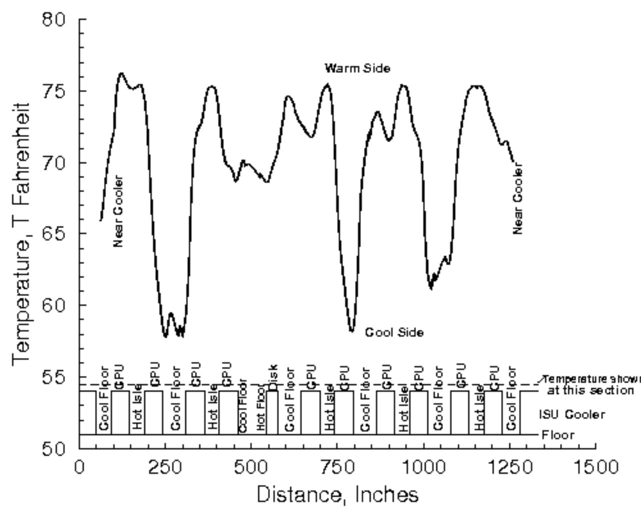

Figure 9. Temperature at a section of 32ft. from the control room wall and at a height of 7 ft.

Figures 10 and 11 show flow visualization of the cooler intake streamlines for the coolers located adjacent to and along the CPU racks (see Fig 1). Fig. 10 shows the streamlines patterns for the existing coolers with both front and top air intakes, while Fig. 11 shows the streamline patterns for the modified coolers with a closed front but enlarged top. The qualitative observation from the flow visualization of the streamline patterns shows about 5% more streamlines are injested into the CPU fan cooler inlet with the modified ISU cooler.

Conclusions and Future Work

The Navier-Stokes equations have been used to model the temperature and velocity fields within the NAS computer room for the Columbia installation. The geometric model for this complex room is scripted so that the various components can be moved around easily and the grids generated in an automated manner.

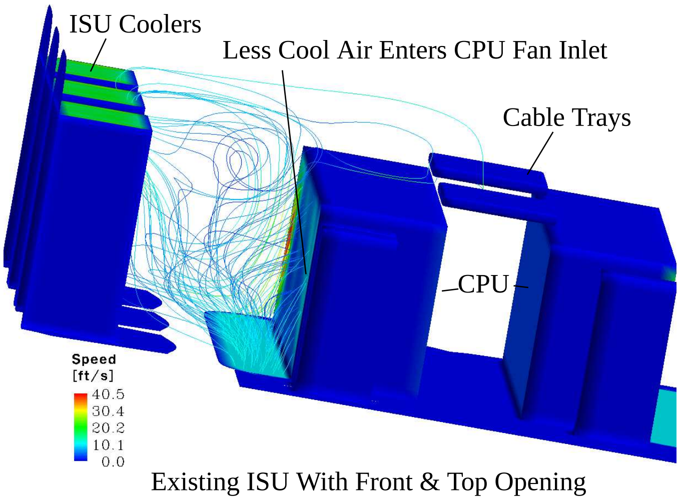

Figure 10. Streamlines from the floor tiles to the nearby existing ISU cooler. Air intake from front and top.

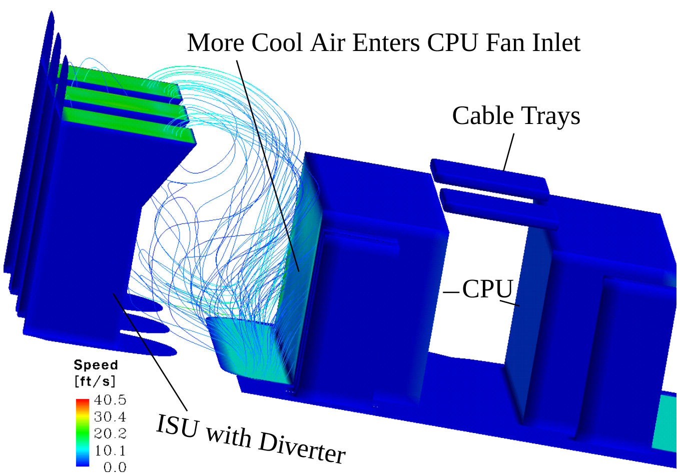

Figure 11. Streamlines from the floor tiles to the nearby proposed modified ISU cooler. Air intake from top only.

The set of boundary conditions used in this simulation appears to work reasonably well. Future simulations will include the subfloor for a more complete analysis. A small amount of short cycling in the CPU rack cooling systems was observed with the current ISU configuration and flow rates. Temperatures were found to be elevated just above the CPU racks, varying between 65-75 degrees Fahrenheit. Modified coolers were modeled and simulated to help assess their effects on nearby CPU fan intakes of cool air. Qualitative flow visualization showed that this ISU modification provided about 5% more cool-air streamlines being ingested by the nearby CPU fan intakes, thus provides better cooling of the CPU racks. A greater improvement might occur when there is more short cycling present, e.g., lower ISU flow rate or ISU failures. Large updrafts on either side of the cable trays and some vortices were observed above some of the racks. Regions of slow air speeds were identified but they did not appear to significantly impact CPU cooling. Some strategically placed ceiling vents or small fans may be able to increase the flow speed, thus improving the cooling and eliminate short cycling. This will be investigated at a future time together with validation of the simulation by comparison with on-site measured data. Failure analysis of one or more ISU coolers and their effects on the overall cooling system will also be explored.

Acknowledgments

Our thanks to Dr. Dochan Kwak and Dr. Walter Brooks of NASA Ames Research Center for their continuing support to this task. Our appreciation goes to Mark Tangney for his valuable help and his sound engineering knowledge of the different aspects of the facility and the air-conditioning system.

Reference

Spalart, P. R., Allmaras, S. R., 1992. A One-Equation Turbulence Model for Aerodynamic Flows. AIAA 92-0439.

Meakin, R. L., 2001. Object X-rays for Cutting Holes in Composite Overset Structured Grids. AIAA 2001-2537, 15th AIAA Computational Fluid Dynamics Conference, June, Anaheim, California.

Schmidt, R. R., Kailash, C. K., Kelker, K. M., Radmehr, A., and Patankar, S. V., 2001. Measurements and Predictions of the Flow Distribution Through Perforated Tiles in Raised-Floor Data Centers. IPACK 2001-15728. Proceedings of IPACK 01, The Pacific Rim/ASME International Electronic Packaging Tech. Conf. and Exh. July 8-13, Kaui, Hawaii, USA, 905-914.

Chan, W. M., 2002. The OVERGRID Interface for Computational Simulations on Overset Grids. AIAA 2002-3188, 32nd AIAA Fluid Dynamics Conference, 24-26 June, St. Louis, Missouri.

Pandya, S. A., Venkateswaran, S., and Pulliam, T. H., 2003. Implementation of Preconditioned Dual-Time Procedures in OVERFLOW. AIAA 2003-0072, 41st Aerospace Sciences Meeting & Exhibit, Jan. 6-9, Reno, NV.

Buning, P. G., Jespersen, D. C., Pulliam, T. H., Chan W. M., Slotnick, J. P., Krist, S. E., and Renze, K. J., 2003. OVERFLOW Users Manual, NASA.

Buning, P. G., Gomez, R. J., and Scallion, W. I., 2004. CFD Approaches for Simulation of Wing-Body Stage Separation. AIAA 2004-4838, 22nd AIAA Applied Aerodynamics Conference, August, Providence, Rhode Island.

Chan, W. M., 2004. Advances in Chimera Grid Tools for Multi-Body Dynamics Simulations and Script Creation. Proceedings of the 7th Symposium on Overset Composite Grid and Solution Technology, 5-7 October, Huntington Beach, California.